CFD SIMULATION OF NATURAL CONVECTION IN OPEN POOL REACTOR COOLING SYSTEM

Rosli Darmawan[1], Nor Mariah Adam[2], Nuraini Abdul Aziz[2] , M Khairol Anuar M Ariffin[2]

[1]*Malaysian Nuclear Agency (NUCLEAR MALAYSIA), Bangi, 43000 Kajang, MALAYSIA.*

[2]*Universiti Putra Malaysia (UPM), 43400 Serdang, Selangor, MALAYSIA*

ABSTRACT

One of the most prevailing issues in the operation of Nuclear Reactor is the safety of the system. Worldwide publicity on a few nuclear accidents as well as the notorious Hiroshima and Nagasaki bombing have always brought about public fear on anything related to nuclear. Most findings on the nuclear reactor accidents are closely related to the reactor cooling system. Thus, the understanding of the behavior of reactor cooling system is very important to ensure the development and improvement on safety can be continuously done. This study attempted to simulate the natural convection cooling inside an open pool research reactor. The study on natural convection with similar configuration using CFD is rarely conducted, therefore, this investigation will help further in understanding the phenomenon. The results reveal the flow regimes, temperature profile and the formation of the natural convection. The model with pure conduction has also been simulated and compared with the natural convection model. The result shows that natural convection could increase heat transfer process in the reactor pool.

ABSTRAK

Salah satu isu yang paling lazim dalam operasi Reaktor Nuklear ialah keselamatan sistem. Publisiti di seluruh dunia mengenai beberapa kemalangan nuklear serta pengeboman Hiroshima dan Nagasaki yang terkenal selalu membawa ketakutan orang ramai terhadap apa-apa yang berkaitan dengan nuklear. Kebanyakan penemuan mengenai kemalangan reaktor nuklear berkait rapat dengan sistem penyejuk reaktor. Oleh itu, pemahaman tentang sistem penyejukan reaktor sangat penting untuk memastikan pembangunan dan peningkatan keselamatan dapat dilakukan secara berterusan. Kajian ini cuba mensimulasikan penyejukan perolakan semulajadi di dalam reaktor penyelidikan kolam terbuka. Kajian mengenai konveksi semulajadi dengan konfigurasi yang sama menggunakan CFD jarang dilakukan, oleh itu, siasatan ini akan membantu lebih jauh dalam memahami fenomena tersebut. Hasilnya mendedahkan aliran aliran, profil suhu dan pembentukan konveksi semulajadi. Model dengan konduksi tulen juga telah disimulasikan dan dibandingkan dengan model perolakan semula jadi. Hasilnya menunjukkan bahawa konveksi semulajadi boleh meningkatkan proses pemindahan haba di kolam reaktor.

Keywords: Natural convection, CFD simulation, nuclear reactor safety

Keywords- CFD simulation, reactor safety, natural convection

INTRODUCTION

The latest nuclear reactor accident on Fukushima Daiichi has drawn worldwide concern on the safety and reliability of the all operating reactors regardless of whether it is nuclear power reactor or nuclear research reactor. All nuclear operators and regulators undertook revision exercises on all aspects of the reactor technology such as the design, safety system and the operating regulation (NEA, 2012). Any effort toward the improvement of reactor safety would at least bring positive perception among the public on the viability of this technology.

The design and development of a nuclear reactor has always been focused on its reliability to operate safely throughout its lifetime. Most of the analysis employed experimental technique before using 1D system level computational codes designed specifically for nuclear reactor design analysis (Yadigaroglu, 2005; Adreani et. al., 2008). Computed Fluid Dynamics (CFD) codes such as FLUENT and CFX have started to be used in addition to 1D simulation codes. The 3D capability is needed in certain cases where 1D code might not be able to predict. Most of the applications are in the Pressurized Water Reactors (PWR) which has more complex design and operating condition compared to Light Water Reactors (LWR). CFD code was used by Anglart and Nylun (1996) and Koncar et. al. (2005) on the study of subcooled boiling phenomenon in PWR reactors. Various other operational conditions in PWR reactors such as slugging, counter current flow limitation, and pressurized thermal shock were simulated by Derdalianto et. al. (2011). Although the operation and design of LWR reactors are less complicated, CFD code is still needed in certain cases. Most of the works in LWR reactors employing CFD method have to do with the emergency analysis of the reactors (Bousbia-salah et. al., 2006). CFD code is also used to review new reactor design. MYRRHA, a Multipurpose hybrid Research Reactor for High-tech Application is being designed in Belgian Nuclear Research Center (SCK-CEN). FLUENT CFD code has been used to model the flow pattern and temperature distribution inside the reactor vessel to improve the design (Vanderhaegen et. al., 2011).

Among the important parts of a reactor design is its cooling system. The decay heat from the reactor core has to be removed effectively to ensure the core temperature remains below melting temperature. Most of the nuclear reactors regardless of its types and technology would keep the reactor core submerged or surrounded by the cooling medium such as water, heavy water or liquid metal, to remove the heat. The study on the dynamics of the heat and mass transfer may involve the understanding of the natural convection phenomenon inside the reactor pool or vessel. Most of the reactor pools for research reactors and reactor vessel for nuclear power plant are of cylindrical shape. The heat source comes from the reactor core located near the bottom in the middle of the cylinder. However, most of the studies on natural convection involves rectangular enclosure and very few studies done on cylindrical enclosures (Rodriguez et.al., 2009). Most of the works on cylindrical shape deal with the horizontal cylinder since its vast applications in piping system. Among the recent studies are experiments and simulations of natural convection heat transfer on horizontal concentric cylinders which help to improve the performance in domestic refrigerators (Atayılmaz, 2011) and underground radioactive waste storage (Webb et. al., 2003). Gustafsson et. al. (2010) used 3D steady state CFD model to study different configurations of piping arrangement to obtain optimum system in a water filled boreholes heat exchanger. Quantino (2012) investigated the effect of perturbation on heat transfer coefficient in heated horizontal cylinder and Hmouda (2010) investigated the heat transfer performance of flue gas piping inside a storage tank for heat reclamation system.

Some of the works on vertical enclosures studied the behavior of various containers subject to the heat sources from outside of the cylinder instead of inside. Oryan (2000), Novak & Nowak (1993) and Heinrich (2003) studied natural convection heat transfer behavior inside rectangular enclosures subject to heating from side and bottom walls. Their investigation focused on flow regimes and localized heating performance inside the enclosures. The process of cooling down and stratifying of the fluid inside vertical cylinder was investigated by Lin and Armfield (1999) and Rodriguez (2009) to understand its behavior. Other works studied the natural convection and evaporation behavior (Khelifi-Touhami et. al., 2010), particles transport phenomena (Akbar et. al., 2009; Pallares & Grau, 2012) and the impact of permeability on stability of buoyancy-opposed flow (Bera and Khalili, 2007) inside vertical containers. There are also studies on natural convection phenomena inside vertical enclosures in building double skin glass facades to understand the heat transfer process, flow regimes, radiation and turbulence effect as well as façades configurations for optimal building design (Pasut and Carli, 2012; Coussirat et. al., 2008). Considering the lack of CFD simulation studies on the natural convection behavior with similar configuration as in reactor cooling pool, an attempt to simulate the natural convection phenomenon in an open pool reactor using CFD simulation code was conducted to understand the thermal behavior and dynamics of the coolant fluid inside the reactor pool cooling system.

THE REACTOR POOL MODEL

The reactor pool model configuration under consideration is shown in Figure 1. It consists of a cylindrical pool with width and height ratio of 1:3. The heat source comes from a cylindrical core located near the bottom, in the middle of the cylinder. The dimensions of the model are derived from an actual open pool research reactor of TRIGA type (Nuclear Malaysia, 2008). The vertical and bottom walls are considered in adiabatic state as the actual reactor pool surrounded by a thick (2.5 meter) of high density concrete. The cylindrical heat source represents the reactor core, elevated at certain height as per actual core condition. The elevation structure is ignored in this model to simplify the simulation and to reduce the computational load. The model is assumed to be in steady state condition to simulate a normal operation of the reactor. The cooling fluid is normal water operating at room temperature and at atmospheric pressure. No external force or pressure is imposed on the fluid, thus it is assumed to be in stationary and laminar flow condition. The pool wall is considered smooth and no frictional force is exerted on the fluid. The TRIGA type reactor pool for research reactors are mostly open pool reactors (Fouquet et. al.,2003). Thus, the top part of the model is considered as convective wall exposed to free stream air at room temperature. The convective heat transfer coefficient in free stream air is set to 10W/m2 K as used by Rodriguez et. al. (2009) and Gastelurutia et. al. (2011). Other fluid properties of the model are shown in Figure 1. The core is subjected to varying levels of heat flux to study the behavior of natural convection in the reactor pool.

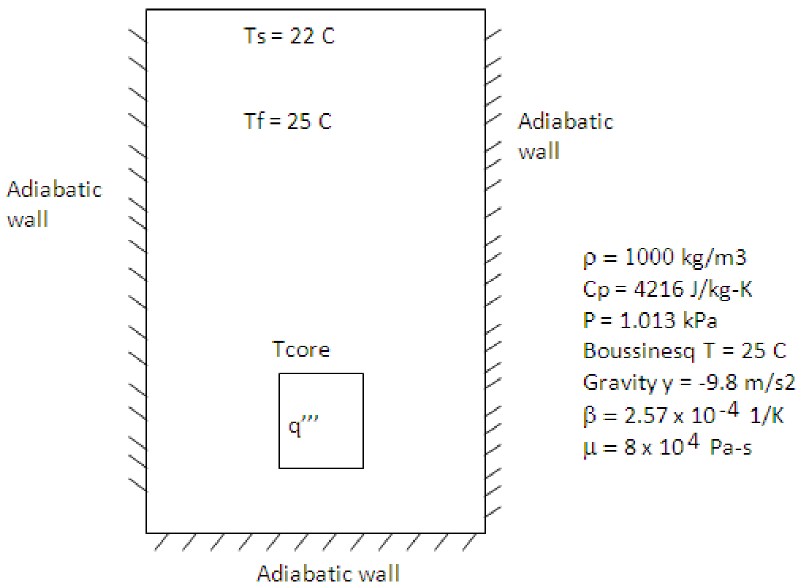


Figure 1: Schematic of the reactor pool model

GOVERNING EQUATIONS

The hydrodynamic and thermal behaviors of the open pool reactor model are solved numerically using CFD computational code FLUENT V6.3. FLUENT is a state-of-the-art CFD code that is widely used in industrial and academic researches for many years worldwide. FLUENT employs finite volume method in solving the algebraic equations through integration of the governing equations about each control volume, and then generates discretization of the equations which conserve each quantity based on control volume ([a]ANSYS, 2006). FLUENT uses the fundamental conservation equations for fluid flow, such as Navier–Stokes for momentum, and the continuity and energy equations, to solve the velocity and temperature fields. Near-wall behaviors of the velocity and temperature profiles are often approximated by wall functions, which rely on the universal behavior of these profiles in the near-wall region ([b]ANSYS, 2006).

The Reynolds Averaged Navier–Stokes (RANS) equations solved by FLUENT are given below assuming no sources, no external forces, no species diffusion, and no viscous heating:

Conservation of Mass (Continuity)

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \qquad (1)$$

where is density and v is the velocity.

Conservation of Momentum (Navier–Stokes)

$$\frac{\partial \rho_f \bar{v}}{\partial t} + \nabla \cdot (\rho_f \bar{v}\bar{v}) = -\nabla p + \nabla \cdot \tau \quad + \rho \vec{g} \qquad (2)$$

where p is the static pressure, is the stress tensor, and g is the gravitational body force. The stress tensor is defined as the following:

$$\tau = \left(\mu(\nabla \bar{v} + \nabla \bar{v}^T) - \frac{2}{3} \nabla \cdot \bar{v} I \right) \qquad (3)$$

where μ is the molecular viscosity, I is the unit tensor, and the second term on the right hand side is the effect of volume dilation.

Conservation of Energy

$$\frac{\partial \rho_f E}{\partial t} + \nabla \cdot (\bar{v}(\rho_f E + p)) = \nabla \cdot (k_f \nabla T + (\tau \cdot \bar{v})) + q''' \qquad (4)$$

where k_f is the effective thermal conductivity, E is total energy (enthalpy plus kinetic energy) and T is temperature. The first two terms on the right-hand side of equation (4) above represent energy transfer due to conduction and viscous dissipation, respectively. Whereas q''' is the heat addition from fuel heating. In these equations, the f subscripts refer to fluid which is water in this case.

In Equation (4), E is defined as the following:

$$E = h - \frac{p}{\rho} + \frac{v^2}{2} \qquad (5)$$

The modeling of natural convection by CFD simulations typically uses Boussinesq approximation model for the buoyancy term to achieve faster convergence with reasonably good predictions (Rodriguez et.al., 2009; Webb et. al., 2003; Novak and Nowak, 1993, Lin and Armfield, 1999). Thus, the term under momentum equation is defined as the following:

$$(\rho - \rho_0)g \approx -\rho_0 \beta (T - T_0) g \qquad (6)$$

This model treats density as a constant value in all solved equations, except for the buoyancy term in the momentum equation, where $_0$ is the constant density of the flow, To is the operating temperature, and is the thermal expansion coefficient.

Equation (6) is obtained by using the the following Boussinesq approximation equation to eliminate from the buoyancy term.

$$\rho = \rho_0(1 - \beta \Delta T) \qquad (7)$$

This approximation is accurate as long as changes in actual density are small; specifically, the Boussinesq approximation is valid as the following.

$$\beta(T - T_0) \ll 1 \qquad (8)$$

Therefore, as long as the product of thermal expansion coefficient and the temperature difference is much less than unity, the above approximation is valid.

MODEL VALIDATION

The model has been validated on two aspects that are the modeling and numerical solutions verification. The modeling verification is to identify any programming errors during modeling process while numerical solution verification is to account for any quantification errors due to computational sources such as grid spacing and convergence criteria.

The model has been verified by comparing the results of known benchmark cases from literatures with that of the model for this case. Although there are no specific benchmark cases that resemble the case under study, several parameters of the model such as temperature and velocity profile were compared for verification purposes. Other works which have almost similar profile of this model have also been used to compare the natural convection modeling results. For numerical solution verification, a grid independent study was conducted. Several schemes of the grid spacing were imposed on the model to verify its independence. The grid scheme ranges from as small as 0.5 of unit grid size up to 3.0 units grid size. The results on temperature, heat transfer coefficient and Nusselt number were compared to arrive at the right size of meshes which produce reasonably independent results. The convergence profiles have also been observed for each scheme to get the least expensive computational cost against the most reliable results.

RESULTS AND DISCUSSION

Grid Independent Study

The grid independent study conducted on the created model shows that the results of heat transfer coefficient and Nusselt numbers begin to be optimized between 1.0 to 2.0 units of grid size, as shown in figure 2 and 3. A further look at the results of temperature profile with different grid schemes shows that the calculation is almost constant at grid scheme between 1.5 and 2.0 units of grid size. Table 1 shows that at 1.5 and 2.0 grid size, the three parameters under consideration were calculated with less than 0.56 % error, which indicates its independent of grid sizes. As shown in figure 4, between 1.5 and 2.0 grid scheme the convergence time is between 50,000 to 100,000 of convergence time. From these two grid schemes, grid scheme of 1.5 units grid size is much more desirable to save the computational time. Thus, this grid size is used to model all of the models under this study.

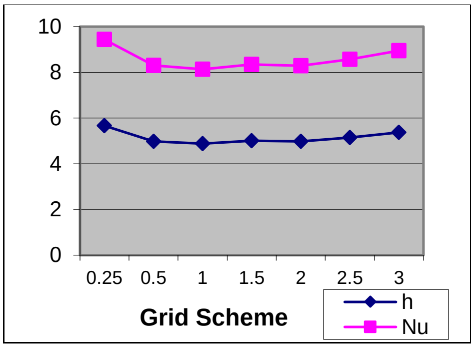


Figure 2: Calculated heat transfer coefficient

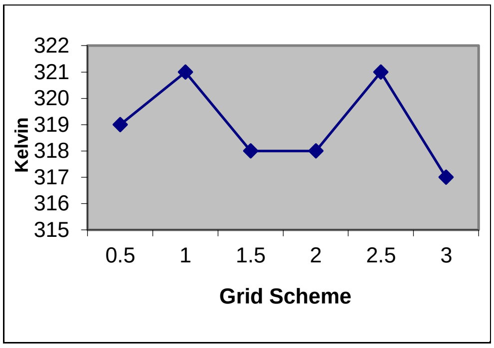


Figure 3: Calculated temperature with different grid schemes and Nusselt number

Table 1: Grid independent study data

Grids Scheme	Tcore K	Error %	h W/m2 K	Error %	Nu	Error %
0.25	314		5.6646		9.4411	
0.5	319	1.59	4.982	12.05	8.3038	12.05
1	321	0.63	4.8814	2.02	8.1357	2.02
1.5	318	0.93	5.0068	2.57	8.3448	2.57
2	318	0.00	4.9789	0.56	8.2982	0.56
2.5	321	0.94	5.1473	3.38	8.5789	3.38
3	317	1.25	5.3685	4.30	8.9475	4.30

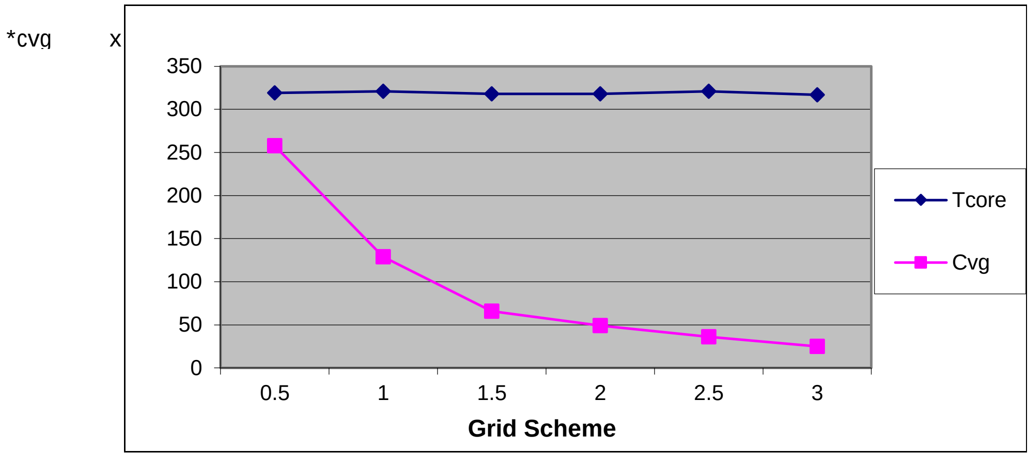


Figure 4: Calculated core temperature and convergence time

Modeling results

The results of the modeling are compared with typical benchmark parameters as shown in figure 5 and figure 6. For temperature profile as shown in figure 6, it shows that the created model have similar trend with typical natural convection case where the temperature reduces as it moves away from the surface. For the velocity similar trend could also be seen on both the model and typical natural convection case. The velocity would rise up to its maximum before reduces as it moves away from the surface.

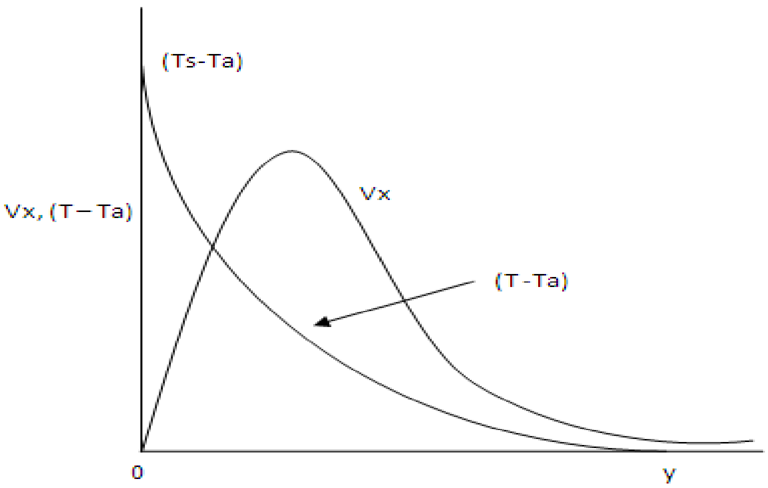


Figure 5: Typical profile of velocity and temperature in natural convection over a surface (Jaluria, 1980).

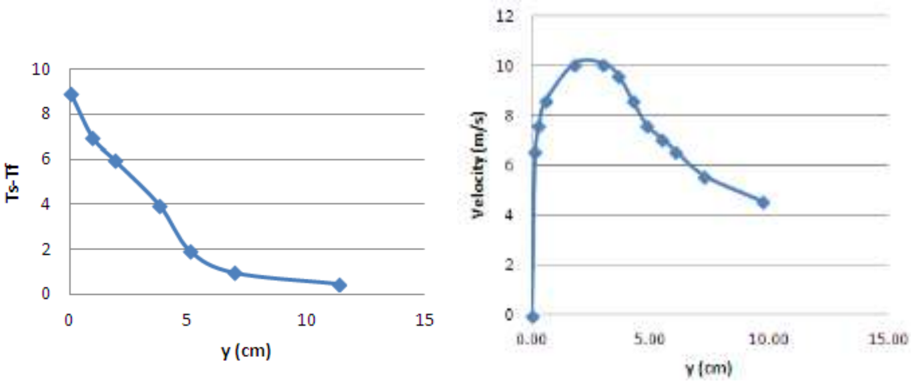


Figure 6: Temperature and velocity profile of the model.

The stream function profiles of this modeling have also been compared with several works on natural convection that have almost similar configuration with the case study. Figure 7 shows that the natural convection circulation occurs near the edges of the heat source. Both profiles of Nakhi et.al.(2008) and the model have similar characteristics where most of the recirculation occurs near the edges due to the high heat transfer rate. This will produce more temperature gradient which induces the buoyancy force that lead to natural convection circulation near the vicinity.

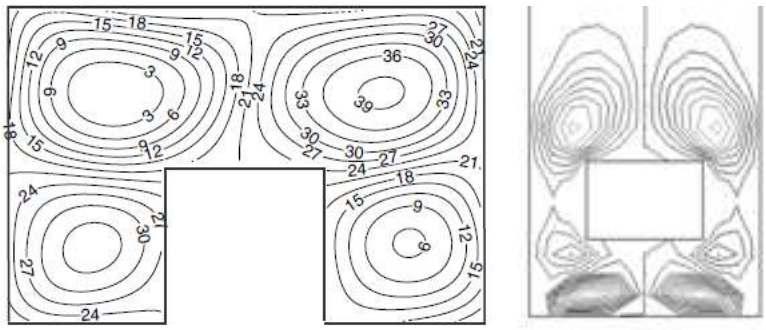


Figure 7: Stream profiles from Nakhi et. al. (2008) on the left side and the reactor pool model on right side.

The overall stream fuction profile of the model is shown in Figure 8. It consists of three significant circulation profiles. This may be due to three different phenomena occurred inside the pool. The bottom part near the heat source, the circulation occurred due to high temperature gradient between the pool water and the core surface. The top part circulation occurred due to temperature gradient between the pool's temperature which may have been raised when mixed with the top stream of bottom circulation. The middle part is considered in between the top and bottom circulations. The natural convection occurred due to the temperature gradient between top part of bottom circulation and bottom part of the top circulation. These phenomena could be compared to the works of Rodriguez (2009) and Dutoit (2006) where both studies found three separate regions formed in either cylinder or rectangular enclosure having heat source from the bottom, once the heat transfer reached a quasi-steady state under transient flow. Figure 8 shows the results from both studies illustrating the bottom region where temperature stratification occurred, followed by middle part, a mixed region of stratification and almost uniform region which dominated the top part of the enclosure.

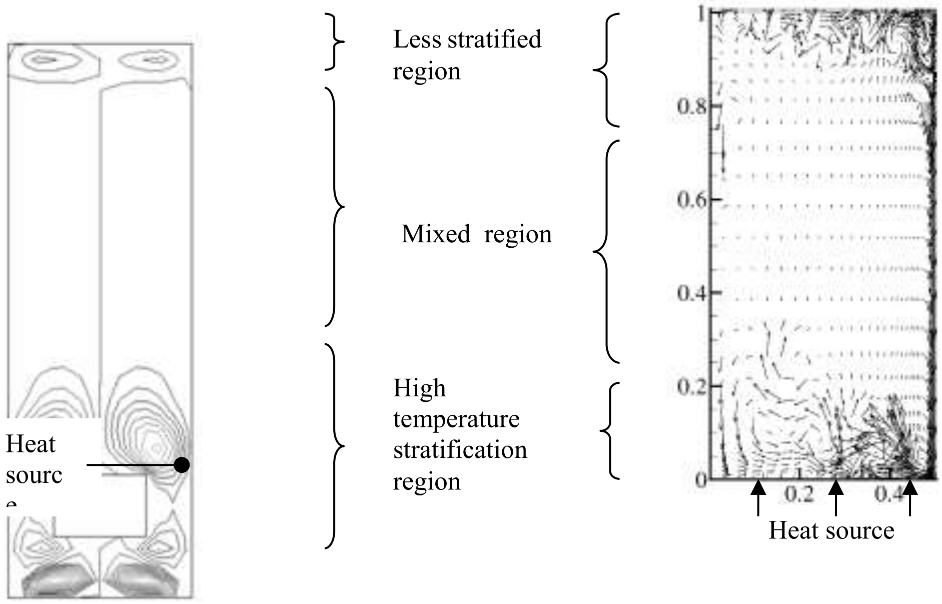
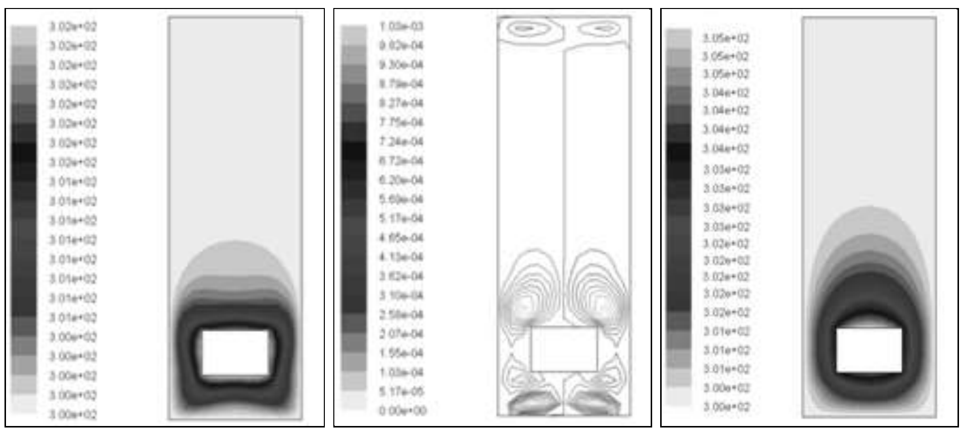


Figure 8: Stream function profiles of reactor pool model (left) and velocity vectors by Rodriguez et. al. (2009) on right hand side.

Natural convection modeling

The modeling of natural convection of the reactor pool using FLUENT 6.3 CFD model was using pressure based solver with Boussinesq approximation scheme as commonly used by many (Rodriguez et.al., 2009; Webb et. al., 2003; Novak and Nowak, 1993, Lin and Armfield, 1999). The results of the modeling with Boussinesq scheme are shown in figure 9a and 9b. The temperature profile shows that the temperature of the core was cooled down to around 29°C and the pool water around the core quickly reached the bulk temperature of 26°C at less than half of the reactor pool height. The profile almost resembles square shape around the core. This is in contrast with the modeling without Boussinesq scheme in figure 9c, where the temperature of the core was cooled down to around 32°C and reached pool water bulk temperature of 26°C at about half of the pool height. The profile resembles oval shape around the core. The model without Boussinesq did not have flow stream while the model with Boussinesq has flow stream as shown in figure 9. The differences between these two models show that with Boussinesq approximation, natural convection occurred which involved both conduction and convection heat transfer, whereas for the model without Boussinesq, only conduction heat transfer occurred. The natural convection shows better cooling effect which cooled down the core surface temperature lower than the pure conduction cooling.



(a) (b) (c)

Figure 9: Temperature profile (a) and flow stream (b) of natural convection simulation as compared to temperature profile without natural convection simulation (c) where no flow contour produced.

Table 2: Comparison between natural convection and pure conduction simulation

Heat flux W/m^2	Natural convec °C	Pure conduct °C
33	29	32
66	34	38
99	38	43
132	41	48
330	62	79
660	97	132
1980	238	344
3300	378	554

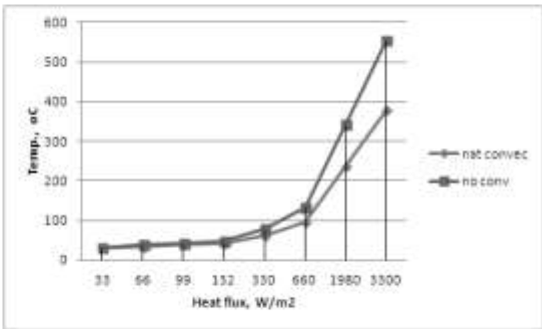


Figure 10: Trend between natural convection and pure convection simulation

The flow stream result shows three regions as discussed earlier. The three regions are divided at the center of the pool due to the natural circulations formed near the edges of the core. The circulation from both edge sides created a boundary at the center of the pool as both streams collided to each other. This collision occurred both at the bottom edge and top edge of the core. The division is continued up to the top of the pool as a result of buoyancy stream from both sides.

Further investigation was conducted to see the effect of increasing heat flux for both models. Table 2 and figure 10 shows the results of the simulation for these two models. The results clearly show the effect of natural convection cooling as compared to pure conduction. All the core surface temperatures with natural convection are cooled down at lower temperatures than without natural convection. At higher heat flux the cooling with natural convection shows even more efficient cooling where the core temperature was cooled down to almost twice that of without natural convection starting at 660 W/m^2 heat flux. The trend can clearly be seen as shown in figure 10, where the differences in core temperature are increasing with increasing heat flux. The results have shown how natural convection was formed and it could increase the heat transfer process compared to pure conduction heat transfer.

CONCLUSION

The design, assessment and investigation of nuclear reactor safety have been traditionally conducted by analytical calculations and by physical experimentations. Later, with the emergence of computing technology, 1D system codes were widely used until the last decade. The lack of detailed predictions especially on component and physical level has attracted the usage of 3D simulation codes to nuclear reactor safety analysis. While CFD codes have been widely used outside the nuclear industry, it is yet to be developed to achieve the necessary credibility in nuclear industry. This paper attempted to simulate a natural convection phenomenon in an open pool reactor using FLUENT 6.3 CFD simulation code. The simulations have shown the characteristics of the natural convection inside the pool as well as its cooling behavior. This simulation would help in understanding the natural convection mechanism in an open pool reactor or similar facilities.

REFERENCES

Akbar, M.K., Rahman, M. and Ghiaasiaan, S.M. (2009). Particle transport in a small square enclosure in laminar natural convection. *Aerosol Science*, 40 (2009), p747-761. doi:10.1016/j.jaerosci.2009.04.007

Andreani, M., Haller, K., Heitsch, M., Hemström, B., Karppinen, I., Macek, J., Schmid, J., Paillere, H., Toth, I. (2008). Benchmarking of CFD Codes for Application to Nuclear Reactor Safety. *Nuclear Engineering and Design*, Volume 238, Issue 3, March 2008, Pages 502-513. doi:10.1016/j.nucengdes.2007.01.021

Anglart, H. and Nylun, O,. (1996)dCFD application to prediction of void distribution in two-phase bubbly flows in rod bundles. *Nuclear Engineering and Design* 163 (1996) p81-98. Elsevier-ssdi 0029-5493(95)01160-9.

[a]ANSYS. (2006). Chapter 25: Using the solver, FLUENT V6.3 User Manual. Fluent Inc.

[b]ANSYS. (2006). Chapter 13: Modeling Heat Transfer, FLUENT V6.3 User Manual. Fluent Inc.

Atayilmaz, S.O. (2011). Experimental and numerical study of natural convectionheat transfer from horizontal concentric cylinders, *International Journal of Thermal Sciences*, 50 (2011) p1472-1483. doi:10.1016/j.ijthermalsci.2011.03.019

Bera, P. and Khalili, A. (2007). Stability of buoyancy opposed mixed convection in a vertical channel and its dependence on permeability. *Advances in Water Resources* 30 (2007), p2296–2308. doi:10.1016/j.advwatres.2007.05.003

Bousbia-salah, A.,Brahim Meftah , Tewfik Hamidouche, El Khider Si-Ahmed. (2006). A Model for the analysis of loss of decay heat removal during loss of coolant accident (LOCA) in MTR pool type research reactor. *Annals of Nuclear Energy* 33 (2006) 405-414 Elsevier. doi: 10.1016/j.anucene.2005.12.001

Coussirat, M., Guardo, A., Jou, E., Egusquiza, E., Cuerva, E., Alavedra, P. (2008). Performance and influence of numerical sub-models on the CFD simulation of free and forced convection in double-glazed ventilated facades, *Energy and Buildings,* 40 (2008) p1781–1789. doi:10.1016/j.enbuild.2008.03.009

Deendarlianto, Höhne, T., Apanasevich, P., Lucas, D., Vallée, C., & Beyer, M. (2012). Application of a new drag coefficient model at CFD-simulations on free surface flows relevant for the nuclear reactor safety analysis. *Annals of Nuclear Energy, 39*(1), 70-82. doi:10.1016/j.anucene.2011.09.010

Fouquet, D.M., Razvi, J., Whittemore, W.L. (2003). TRIGA research reactors: A pathway to the peaceful applications of nuclear energy, *Nuclear News*, November 2003 p46-56. General Atomics, San Diego, California.

Gastelurrutia,J., Ramos, J.C., Larraona, G.S., Rivas, A., Izagirre, J., Luis del Río (2011). Numerical modelling of natural convection of oil inside distribution transformers, *Applied Thermal Engineering* 31 (2011) p493-505. doi:10.1016/j.applthermaleng.2010.10.004

Gustafsson,A.M., Westerlund,L., Hellstrom, G. (2010). CFD-modelling of natural convection in a groundwater-filled borehole heat exchanger, *Applied Thermal Engineering*, 30 (2010) p683–691. doi:10.1016/j.applthermaleng.2009.11.016

Heinrich, M. (2003). Numerical simulation of heat transfer by natural convection in cavities of facade elements, *Energy and Buildings*, 35 (2003) 305–311. PII: S0 378-7788(02)0008-9

Hmouda, I., Rodriguez, I., Bouden, C., Oliva, A. (2010). Unsteady natural convection cooling of a water storage tank with an internal gas flue, *International Journal of Thermal Sciences,* 49 (2010) p36–47. doi:10.1016/j.ijthermalsci.2009.05.011

Jaluria, Y. (1980). Natural convection – HMT, the science and applications of heat and mass transfer, vol. 5. Pergamon Press Ltd, Oxford, England. ISBN 0-08-025432-2.

Khelifi-Touhami, M.S., Benbrik, A., Lemonnier, D. and Blay, D. (2010). Laminar natural convection flow in a cylindrical cavity application to the storage of LNG. *Journal of Petroleum Science and Engineering,* 71 (2010), p126–132. doi:10.1016/j.petrol.2009.12.011

Koncar, B., Krepper, E., Egorov, Y. (2005). CFD Modelling of Subcooled Flow Boiling for Nuclear Engineering Application. International Conference Nuclear Energy for New Europe 2005 Bled, Slovenia, September 5-8, 2005. p140.1-140.14

Lin, W. X. and Armfield, S. W. (1999). Direct Simulation of natural convection cooling in a vertical cylinder, *International Journal of Heat and Mass transfer* 42 (1999), p4117-4130. PII: S0017-9310(99)00074-5

NEA. (2012). Actions taken by regulatory bodies and international organizations following the Fukushima Daiichi nuclear accident, *Nuclear Safety, September 2012.* Nuclear Energy Agency (NEA), Organisation for Economic Co-operation and Development (OECD), Paris, France.

Novak, M.H. and Nowak, E.S. (1993). The CAV program for numerical evaluation of laminar natural convection heat transfer in vertical rectangular cavities. *Computer Physics Communications,* 78 (1993), p95-104. doi:0010-4655/93

Nuklear Malaysia. (2008). Chapter 5: Reactor System, *RTP Safety Analysis Report.* Agensi Nuklear Malaysia. 2008.

Oryan, A. and Yang, W. J. (2000). Natural Convection in Enclosures with Localized Heating from Below and Symmetrical Cooling from Sides. *International Journal of Numerical Methods for Heat & Fluid Flow*, Vol 10, No 5, 2000. Pp518-529. MCB University Press, 0961-5539.

Pallares, J and Grau, F.X. (2012). Particle dispersion in a turbulent natural convection channel flow. *Journal of Aerosol Science*, 43 (2012). p45-46. doi:10.1016/j.jaerosci.2011.09.002

Pasut, W. and Carli, M.D. (2012). Evaluation of various CFD modelling strategies in predicting airflow and temperature in a naturally ventilated double skin facade, *Applied Thermal Engineering,* 37 (2012) p267-274. doi:10.1016/j.applthermaleng.2011.11.028

Quintino, A. (2012). Experimental analysis of the heat transfer coefficient for a heated cylinder in cross-flow downstream of a grid flow perturbation. *Applied Thermal Engineering,* No 35 (2012). p55-59.

Rodríguez, I., Castro, J., Pérez-Segarra, C.D., Oliva, A. (2009). Unsteady numerical simulation of the cooling process of vertical storage tanks under laminar natural convection, *International Journal of Thermal Sciences* 48 (2009) p708–721. doi:10.1016/j.ijthermalsci.2008.06.002

Vanderhaegen, M., Vierendeels, J., Arien, B. (2011). CFD analysis of the MYRRHA primary cooling system. *Nuclear Engineering and Design*, Volume 241, Issue 3, March 2011, Pages 775-784. doi:10.1016/j.nucengdes.2010.12.009

Webb W.S., Francis, N.D., Dunn, S.D., Itamura, M.T. and James, D.L. (2003). Thermally induced natural convection effects in Yucca Mountain drifts. *Journal of Contaminant Hydrology* 62-63 (2003). p713-730. doi:10.1016/S0169-7722(02)00180-8

Yadigaroglu, G. (2005). Computational Fluid Dynamics for nuclear applications: from CFD to multi-scale CMFD. *Nuclear Engineering and Design* 235 (2005) 153-164. Elsevier doi:10.1016/j.nucengdes.2004.08.044